

# Modeling of Spray Cooling: Effect of Droplet Velocity and Liquid to Vapor Density Ratio on Heat Transfer

R. Panneer Selvam<sup>1</sup>, Mita Sarkar<sup>1</sup>, and Rengasamy Ponnappan<sup>2</sup>

<sup>1</sup>Bell 4190 University of Arkansas, Fayetteville, AR 7270,

Ph: 479-575-5356, email: rps@engr.uark.edu,

<sup>2</sup>Air Force Research Laboratory, Propulsion Directorate, Wright-Patterson AFB, OH 45433-7251

## Abstract

Numerical modeling of multiphase flow using level set method is discussed. The 2-D model considers the effects of surface tension between liquid and vapor, gravity, phase change and viscosity. The level set method is used to capture the movement of the free surface. The details of incorporating the mechanism of phase change in the incompressible Navier-Stokes equations using this method is described. The governing equations are solved using the finite difference method. The viability of using different nondimensional Navier-Stokes equation is investigated. The computer model is used to study the spray cooling phenomenon in the micro environment of about 40  $\mu\text{m}$  thick liquid layers with vapor bubble growing due to nucleation. The effect of velocity and density ratio variations on heat transfer is investigated systematically for the case of droplet impact on vapor bubble.

## 1. Introduction

Spray cooling is a high flux heat removal technique considered for high power systems such as advanced lasers and power switches. The spray cooling with phase change and droplet impact can achieve heat fluxes up to  $1000 \text{ W/cm}^2$  as reported by Yang et al.(1996). Several experiments have been conducted using spray cooling in recent years (Chow et al. 1997, Lin and Ponnappan 2003, and Mudawar, 2001) and various designs of spray cooling devices are emerging. Theoretical understanding of the spray cooling heat acquisition phenomena is still in its infancy and a focused effort to develop a comprehensive numerical model is a prime importance to this field.

Even though spray cooling has been used in the industry for several years the overall theoretical understanding is limited due to complex interaction of liquid, vapor, liquid droplet impact and phase change as sketched in Fig.1. Only limited work is available in the related area of bubble dynamics, pool boiling, droplet impact on a hot plate and some simplified model for spray cooling. A detailed survey on current status of computer modeling of spray cooling and methods to solve multiphase flow is presented in Selvam et al. (2005). Recently Selvam and Ponnappan (2004), and Selvam et al. (2005) identified that computer modeling of nucleation boiling in thin film in the neighborhood of 70  $\mu\text{m}$  including droplet impact will provide valuable information in the design of experiments for spray cooling. Preliminary computation of a growing of vapor bubble in thin film of liquid and the transient heat transfer on the wall are reported by Selvam et al.(2005a). Later the heat transfer due to bubble growing and bursting in to vapor and the impact of droplet on bubble is reported in Selvam et al. (2005b & c). In these works the droplet velocity

is kept to be 2.55 m/s and the density ratio is kept to be 20 due to computational limitation. To apply the computer model to various density ratios and droplet velocities, a new non-dimensional equation is suggested in this work. Using this new model, further work on computer modeling of the effect of droplet velocity and density ratio on heat transfer is reported. The heat transfer rate at different instants for each case is presented.

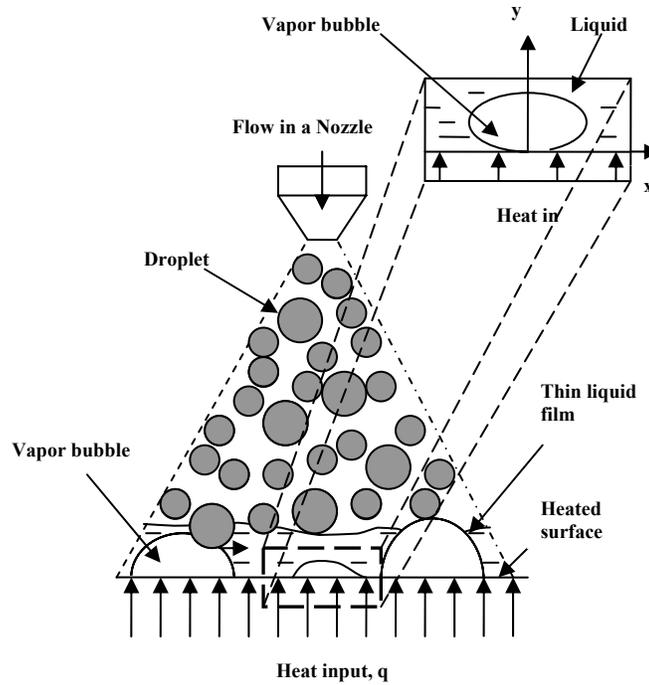


Fig.1. Spray cooling phenomena

## 2. Numerical formulation for multiphase flow using level set method

For a survey on numerical techniques used to model multiphase flow and their advantages and disadvantages one can refer to the literature (Selvam et al. 2005). Here, for computer modeling of liquid and vapor during nucleate boiling, the level set method introduced by Sussman et al. (1994) for bubble dynamics which was modified by Son and Dhir (1998) to accommodate the effect of phase change is used. The interface separating the two phases is captured by a function  $\phi$  which is defined as a positive or negative distance from the interface. Similar to Son and Dhir (1998) and Son et al. (2002) the negative sign is chosen for the vapor phase and the positive sign is chosen for the liquid phase. For more details on the level set method and its application one can refer to Sethian (1999) and Osher and Fedikiw (2003). The extensive application of the level set method in various areas of science and engineering are illustrated with their basic development in the above two books.

## 2.1 Governing equations

In the present model, the fluid properties including density, viscosity and thermal conductivity are constant in each phase and the flow is assumed to be incompressible. The Navier- Stokes equations considering the effect of surface tension, gravity and phase change at the interface are as follows:

$$\rho(\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \rho \mathbf{g} - \sigma \kappa \nabla H + \nabla \cdot \mu \nabla \mathbf{u} + \nabla \cdot \mu \nabla \mathbf{u}^T \quad (1)$$

$$\rho c_{pl}(\partial_t T + \mathbf{u} \cdot \nabla T) = \nabla \cdot k \nabla T \quad \text{for } H > 0 \text{ and } T = T_{\text{sat}}(p_v) \text{ for } H = 0 \quad (2)$$

$$\nabla \cdot \mathbf{u} = \mathbf{m} \cdot \nabla \rho / \rho^2 \quad (3)$$

$$\text{where : } \rho = \rho_v + (\rho_l - \rho_v)H. \quad (4)$$

The value of  $\mu$  and  $k$  are calculated using the similar relation in Eq. (4). Here:

$$\begin{aligned} H &= 1 \text{ if } \varphi \geq 1.5h \\ &= 0 \text{ if } \varphi \leq -1.5h \\ &= 0.5 + \varphi / (3h) + \sin[2\pi \varphi / (3h)] / (2\pi) \text{ if } |\varphi| \leq 1.5h \end{aligned} \quad (5)$$

Where  $h$  is a grid spacing. The Eq. (5) implies that the interface separating two phases is replaced by a transition region of finite thickness. The volume source term included in the continuity equation (3) due to liquid-vapor phase change is derived from the conditions of mass continuity and energy balance at the interface:

$$\mathbf{m} = \rho(\mathbf{u}_{\text{int}} - \mathbf{u}) = k \nabla T / h_{fg} \quad (6)$$

In the level set formulation, the level set function  $\varphi$ , is advanced and reinitialized as:

$$\partial_t \varphi = -\mathbf{u}_{\text{int}} \cdot \nabla \varphi \quad (7)$$

$$\partial_t \varphi = \varphi_0 (1 - |\nabla \varphi|) / \sqrt{(\varphi_0^2 + h^2)} \quad (8)$$

where  $\varphi_0$  is a solution of Eq. (7).

The surface tension effect is considered in the momentum equation by using a step function  $H$  ( $H=0$  in the vapor and 1 in liquid) and  $\kappa$  is the interfacial curvature expressed as:

$$\begin{aligned} \kappa &= \nabla \cdot (\nabla \varphi / |\nabla \varphi|) \\ &= (\varphi_y^2 \varphi_{xx} - 2 \varphi_x \varphi_y \varphi_{xy} + \varphi_x^2 \varphi_{yy}) / (\varphi_x^2 + \varphi_y^2)^{3/2} \text{ for 2D} \end{aligned} \quad (9)$$

Here subscripts are differentiation with respect to  $\varphi$ . The surface tension force,  $-\sigma \kappa \nabla H$  is implemented in the volume form to avoid the need for explicit description of the interface as suggested by Brackbill et al.(1992).

## 2.2 Nondimensional form of the governing equations

The nondimensional form of the above set of equations is derived using the characteristic length  $l_r$ , velocity  $u_r$ , time  $t_r$  and dimensionless temperature  $T^*$ . Length  $l_r$  is taken as the diameter of the droplet and velocity  $u_r$  is taken as the velocity of the droplet. The other two are defined as follows:

$$t_r = l_r / u_r \text{ and } T^* = (T - T_{\text{sat}}) / (T_w - T_{\text{sat}}). \quad (10)$$

In addition, considering  $\rho$ ,  $k$ ,  $\mu$  and  $c_p$  of liquid as reference values, the nondimensional equations without their superscripts are expressed as follows:

$$\rho(\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \rho g_y / Fr^2 - \kappa \nabla H / We + (\nabla \cdot \mu \nabla \mathbf{u} + \nabla \cdot \mu \nabla \mathbf{u}^T) / Re \quad (11)$$

$$\rho c_{pl} (\partial_t T + \mathbf{u} \cdot \nabla T) = (\nabla \cdot k \nabla T) / Pe \quad \text{for } H > 0 \quad (12)$$

$$\nabla \cdot \mathbf{u} = Ja k \nabla T \cdot \nabla \rho / (Pe \rho^2) \quad (13)$$

$$\mathbf{u}_{\text{int}} = \mathbf{u} + Ja k \nabla T / (Pe \rho) \quad (14)$$

where:  $Re = \rho_l u_r l_r / \mu_l$ ,  $We = \rho_l u_r^2 l_r / \sigma$ ,  $Ja = c_{pl} \Delta T / h_{fg}$ ,  $Fr = u_r / \sqrt{g l_r}$ ,  $Pr = c_{pl} \mu_l / k_l$  and  $Pe = Re Pr = \rho_l u_r l_r c_{pl} / k_l$ . Here  $g_y$  represents unit gravitational force in the y-direction. In the Eqs (11) to (14),  $\rho$ ,  $k$ ,  $\mu$  and  $c_p$  are dimensionless with respect to the reference values. In the above equation if the reference velocity is taken other than droplet velocity then the method will be similar to Selvam et al. (2005). The two methods are identified as follows and the advantage and disadvantage are reported.

*Method -1:* In the above equations, if  $l_r$  is taken as the droplet diameter and  $u_r$  is taken as the velocity of the droplet, then  $Re$ ,  $We$ ,  $Pe$  &  $Fr$  needs to be calculated for every droplet velocity.

*Method -2:* If  $u_r$  is taken as some reference value and the droplet velocity in the computational domain is given as the ratio of droplet velocity/ $u_r$ . Then the time step needs to be adjusted with respect to the droplet velocity.

## 2.3 Boundary conditions

The boundary conditions for the governing equations are shown in Fig. 2 and also given below:

At the wall ( $y=0$ ):  $u = v = 0$ ,  $T = T_w$ ,  $\phi_y = 0$ . At the planes of symmetry ( $x=0$  and  $x= x_{\text{max}}$ ):  $u = v_x = T_x = \phi_x = 0$ . At the top of the computational domain (free surface,  $y= y_{\text{max}}$ ):  $u_y = v_y = \phi_y = 0$ ,  $T = T_{\text{sat}}$

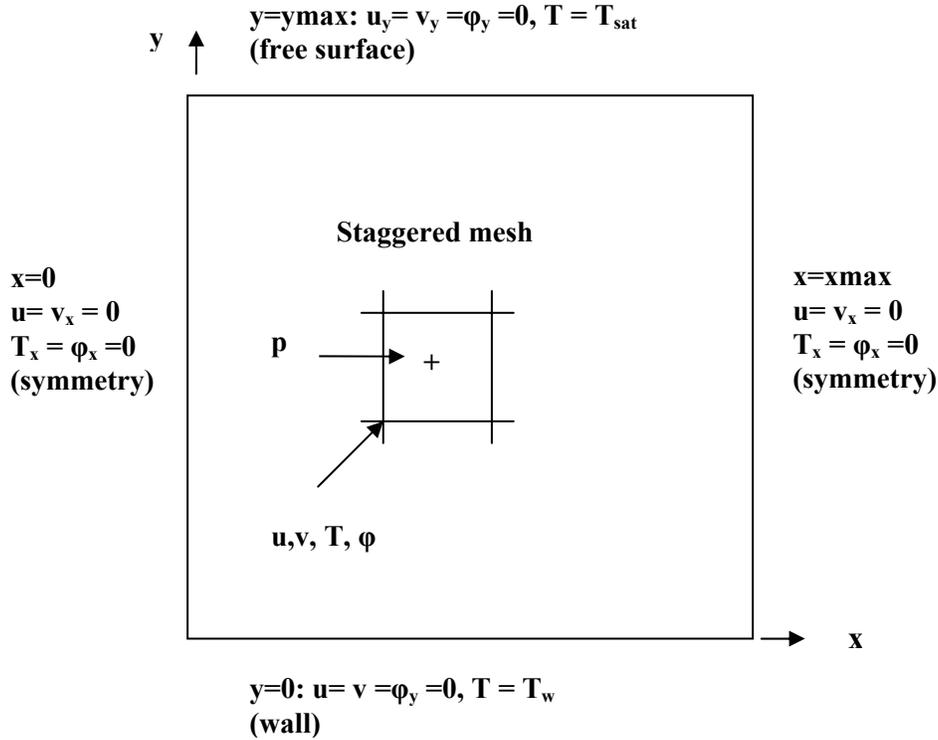


Fig. 2. Boundary conditions and the location of the variables stored in staggered grid system

#### 2.4 Numerical solution

The governing equations Eq. (1), (2), (3), (7) and (8) combined together are highly nonlinear. The equations are discretized using finite difference method on a staggered grid system in which all the variables except pressure are stored at the grid points; and pressure alone is stored at the cell center as shown in Figure 1. The diffusion terms are considered implicitly and the convection and source terms are considered explicitly in time. For spatial approximations all terms are considered using second order central difference and the convection term by a second-order ENO method described by Chang et al.(1996) to prevent numerical oscillations. The pressure and velocity are solved in a sequential manner by the procedure described in Selvam (1997).

The discretized equations from the momentum, energy and pressure equations are symmetric and they are solved by the preconditioned conjugate gradient procedure (Ferziger and Peric, 2002) in an iterative form. The iteration is done until the average residue for each node is reduced to less than  $10^{-9}$ . This amount of accuracy is needed because of high density difference between liquid and vapor. After assuming initial position for distance functions, at each time-step the equations are solved sequentially in the following order:

1. Solve the momentum equations, Eq. (1) for velocities
2. Correct the velocity to take the pressure effect
3. Solve the pressure Poisson equation to satisfy continuity

4. Update the velocities to include the new pressure effect
5. Solve temperature equation Eq. (2)
6. Solve the distance function Eq. (7)
7. Reinitialize the distance function as per Eq. (8) and go to next time step

During the computation, time steps were chosen to satisfy the Courant-Fredreichts-Lewy (CFL) condition,  $\Delta t \leq \min (h/(|u|+|v|), 10^{-6})$ . This was done because of the explicit treatment of the convection terms and the condition that the numerical results should not change if the time steps are halved.

### 3. Results and Discussion

The droplet impact on bubble study conducted by Selvam et al. (2005) is considered here for comparing Method 1 and Method 2 in the previous section. The reference values considered for the computation are: reference length (droplet diameter)  $l_r = 44.172 \mu\text{m}$ , reference velocity (droplet velocity)  $u_r = 2.55\text{m/s}$ , reference time  $t_r = 17.3224 \mu\text{s}$  and  $\Delta T = 10^\circ \text{C}$ . The density ratio of liquid to vapor ( $\rho_l/\rho_v$ ) is 138 and other nondimensional numbers are:  $Re = 393$ ,  $We = 54.4$ ,  $Pe = 3690$ ,  $Ja = 0.127$ ,  $Fr^2 = 1.5 \times 10^4$ . The above data refers to the Lin and Ponnappan (2003) spray cooling experiments using FC-72 for  $T_{\text{sat}} = 53^\circ \text{C}$  case. For the comparison of the two methods  $\rho_l/\rho_v = 20$  is considered which is similar to the previous work reported in Selvam et al. (2005). For Method 2, the reference velocity is considered to be 1m/s and the droplet velocity is still the same value of 2.55m/s. The considered computational domain is 3.3324 units x 3.3324 units which are equal to  $147.2 \mu\text{m} \times 147.2 \mu\text{m}$ . The computational domain is discretized by a 201x201 mesh. The smallest grid size is  $0.7362 \mu\text{m}$ . The time steps used are 86.6 ns ( $5 \times 10^{-3}$  units) for method 1 and 86.6 ns ( $2 \times 10^{-3}$  units) to 22 ns ( $0.5 \times 10^{-3}$  units) for method 2. The time step used by Selvam et al. (2005) is 43.3 ns for the similar case.

#### 3.1 Comparison of Method 1 and Method 2

For the comparison, the droplet impact on a thin liquid film with vapor bubble growing as shown in Fig. 3 is considered. A vapor bubble radius of 0.9167 (40.4925  $\mu\text{m}$ ) units in a liquid layer of 1 (44.172  $\mu\text{m}$ ) units is considered as initial condition. The temperature is assumed to be varying linearly from  $T_w$  to  $T_{\text{sat}}$  from the wall to 0.2 units above the wall. A droplet diameter of 1 (44.172  $\mu\text{m}$ ) units is located at 1.33 (58.75  $\mu\text{m}$ ) units from the left edge and 2.166 (95.68  $\mu\text{m}$ ) units from the bottom. The droplet is falling down with a speed of 1 unit (2.55 m/s) for Method 1 and 2.5 units for Method 2. Since the reference velocity in method 2 is 1m/s, the droplet velocity becomes 2.5 units for the same velocity.

The computed Nusselt number and maximum velocity in the computational domain are plotted for each time in Fig. 4 using both methods. For the similar time step Method 1 velocity plot is much smoother than Method 2. The oscillation using Method 2 is due to numerical instabilities. The maximum velocity in the computational domain is reported to be 13.26 m/s (5.2 units) for Method 1 and 24 m/s (24 units) for Method 2. A run using a smaller time step of 22.08 ns ( $0.5 \times 10^{-3}$  units) which is one fourth of the previous run is conducted. This takes four times the computer time of the 88.34 ns time step. The velocity plot in Fig. 5 is much smoother and the maximum velocity in Fig. 4 is 8.1 m/s which is similar to Method 1. So for droplet impact

studies the new reference velocity and length are advantageous. The method is computationally economical and gives a stable solution. For bubble growth work, the reference values reported in Selvam et al. (2005) needs to be considered. This computer model will be applied for further studies on the effect of density ratio and velocity of droplet impact on heat removal.

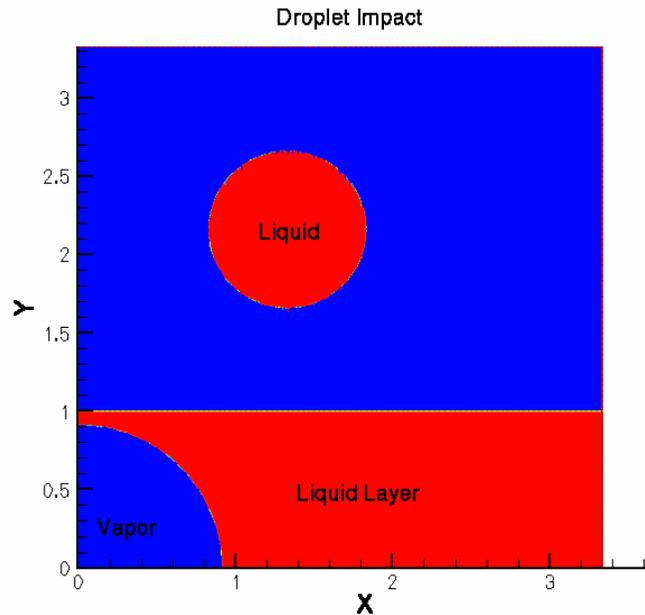


Fig. 3. Computational region and droplet position for droplet impact study

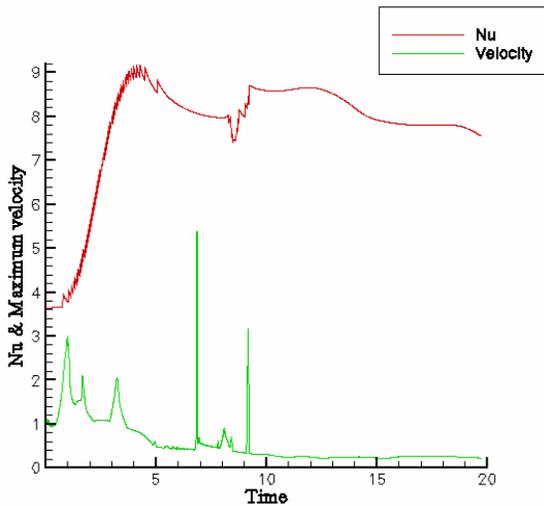


Fig. 4a

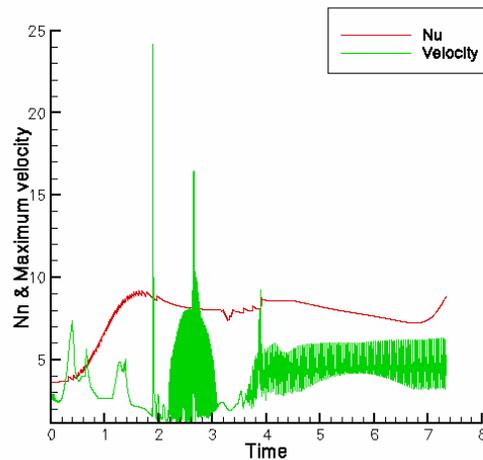


Fig. 4b

Fig. 4. Variation of average Nusselt number and maximum velocity in the computational region with time up to 341  $\mu$ s (4000 time step). (a) Method 1 using a time step of  $5 \times 10^{-3}$  units (86.61 ns) (b) Method 2 using a time step  $2 \times 10^{-3}$  units (88.34 ns).

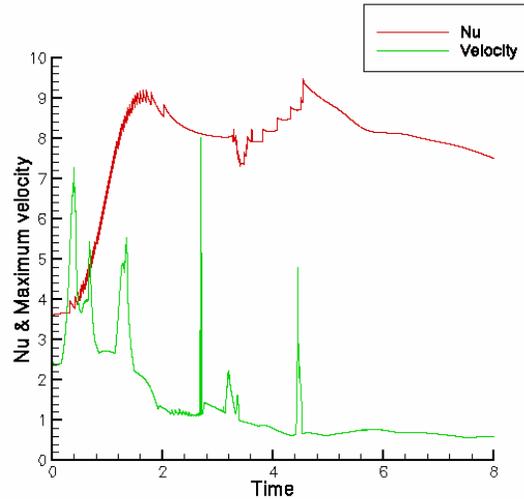


Fig. 5. Variation of average Nusselt number and maximum velocity in the computational region with time up to 353  $\mu$ s (16000 time step) using Method 2. Using a time step  $0.5 \times 10^{-3}$  units (22.08 ns).

### 3.2 Effect of Droplet Velocity on Heat Removal

Using the same parameters as in section 3.1, the velocity of the droplet is varied from 2.55 m/s to 15m/s. The 15m/s is little above the value of 10m/s considered in the experiment by Baysinger et al. (2004). The reference values suggested for Method 1 is used. All the runs were made for about 340  $\mu$ s. The variation of Nusselt number (Nu) with time is similar to the one plotted in Fig. 5 except that for higher velocities the initial slope is much steeper. The details of the variation are reported in Sarkar and Selvam (2005). The maximum Nu for each velocity is reported in Fig. 6. The variation of the Nu with regard to time is gradual. The increase in Nu is 25.6% from 2.55 m/s to 15 m/s. The Nu varied from 9.1 for 2.55m/s to 11 for 15m/s. In Selvam et. al. (2005a), the bubble growth due to nucleation (case 1) and in Selvam et. al. (2005b), bubble bursting when the bubble grows and joins the vapor above the liquid layer (case 2) as well as the bubble bursting due to droplet impact (case 3) are reported. The increase in Nu from bubble growth (case 1) to bubble bursting (case 2) is 120% and bubble bursting (case 2) to droplet impact on bubble (case 3) is 45.45%. The effect of vapor bubble crowding during droplet impact may further increase the Nu as reported in Selvam et al. (2005c). Over all, from the particular study it is concluded that droplet impact has a major effect on Nu. At the same time after some droplet velocity value the variation in velocity has not much influence on Nu.

The maximum Nu for 10m/s is 10.11 which is equivalent to  $12.44 \text{ W/cm}^2$ . Lin and Ponnappan (2003) in their experimental studies reported heat flux in the range of  $70 \text{ W/cm}^2$ . The discrepancy may be due to the fact that, in the actual experiment the droplet impact is 3D whereas here the droplet impact study is 2D. In addition, in the actual spray experiment there will be convective flow over the bubble. In the computational case it is yet to be included. Further work is underway to consider the convective velocity effect on bubble as well as 3D modeling of droplet impact on vapor bubble.

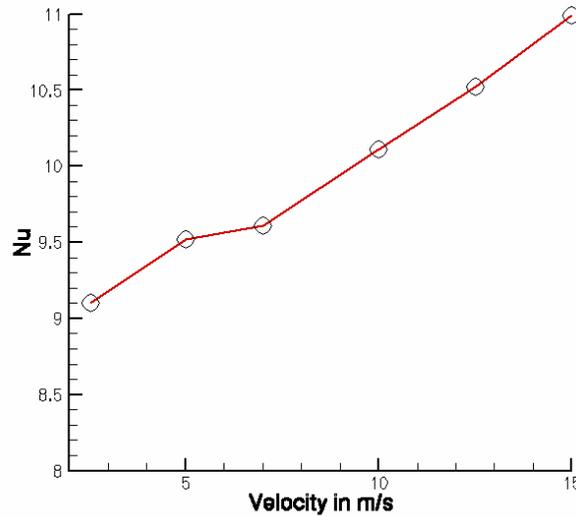


Fig. 6 Variation of Max. Nusselt number with droplet velocity

### 3.3 Effect of Density Ratio on Heat Removal

All the above studies were conducted using liquid to vapor density ratio of 20. In the actual experiment for FC-72 the density ratio is 138. To consider the density effect on heat flux, different values (20 to 120) were considered using the model. The geometry and grid details were the same as before. The droplet velocity considered for this study is 2.55 m/s which is same as section 3.1. Until density ratio of 90, the computer model could run and for 120 the model could not run due to computational instability. Then the model is modified by using a under relaxation factor of 0.7 for pressure. For density ratio of 20, for some runs the vapor bubble did not burst in section 3.1. Whereas, when the density ratio is increased the bubble burst due to droplet impact in all cases. The maximum Nu reached to around 10 for density ratio of 40 and 90 and for density ratio of 120 the value decreased to 8.5. This may be due to computational difficulties at the instant of merging of the droplet with liquid film and the dampening effect of under relaxation. Over all, the effect of density ratio on Nu is not noticeable as much. The plot of Nu verses time for each density ratio is reported in Sarkar and Selvam (2005). Further study using water and other liquids will also be considered in the future.

## 4. Conclusions

Numerical modeling of multiphase flow in spray cooling using level set method is discussed. The model considers the effect of surface tension between liquid and vapor, gravity, phase change and viscosity. The computer model is used to study the spray cooling phenomenon in the micro environment of about 40  $\mu\text{m}$  thickness liquid layers with vapor bubble growing due to nucleation and droplet impact. A new nondimensional equation is suggested for the solution of Navier-Stokes equation which is more stable than the one reported in Selvam et al. (2005). Using

the model the effect of droplet velocity and density ratio on heat flux is investigated. From the study the following conclusions are made:

1. Method 1 is computationally faster and more stable than Method 2.
2. The Nu varied from 8.76 for 2.55m/s to 11 for 15m/s droplet velocity, an increase of 25.6% in Nu; whereas the increase in Nu from bubble growth (case 1) to bubble bursting (case 2) is 120% and bubble bursting (case 2) to bubble bursting due to droplet impact (case 3) is 45.45% as reported in Selvam et al. (2005b).
3. The effect of density ratio on Nu is very little. For low density ratio the bubble did not burst due to droplet impact and for high density ratios always the bubble burst due to droplet impact.
4. The computed heat flux due to droplet impact is in the range of 12.44 W/cm<sup>2</sup>; where as the experimental observations were in the range of 70 W/cm<sup>2</sup>. This discrepancy may be due to not considering the 3D effect of droplet impact on vapor bubble and the convective flow in the liquid film. These effects need to be considered in the future.

## Acknowledgments

This work was conducted partly at the AFRL/WPAFB Propulsion Directorate's Power Division, Thermal Laboratory during the summer of 2005. The first author acknowledges the support received from the US Air Force Research Lab through the American Society of Engineering Education for summer faculty program and through Universal Technology Corporation research grant F33615-02-D-2299 and the ONR/DEPSCoR grant through the University of Arkansas to perform this work and the encouragement provided by Dr. Juan Balda, Dr. Fred Barlow and Dr. Aicha Elshabini, Department of Electrical Engineering, University of Arkansas to visit WPAFB during the summer of 2005.

## References

1. K.M. Baysinger et al. (2004), Design of a microgravity spray cooling experiment, *AIAA-2004-0966*, 42<sup>nd</sup> AIAA Aerospace Sciences Conference and Exhibit, Jan. 5-8, Reno, NV
2. J.U. Brackbill, D.B. Kothe and C. Zang (1992), A continuum method for modeling surface tension, *J. Comp. Physics*, 100, 335-354.
3. Y.C. Chang, T.Y. Hou, B. Merriman and S. Osher (1996), A level set formulation of Eulerian interface capturing methods for incompressible fluid flows, *J. Comp. Physics*, 124, 449-464.
4. L.C. Chow, M.S. Sehmey and M.R. Paris (1997), High heat flux spray cooling, *Ann. Rev. Heat Transfer*, 8, 291-318
5. J.H. Ferziger and M. Peric (2002), *Computational Methods for Fluid Dynamics*, Springer, New York
6. L. Lin and R. Ponnappan, (2003), Heat transfer characteristics of spray cooling in a close loop, *Int. J. Heat Mass Transfer*, 46, 3737-3746
7. I. Mudawar (2001), Assessment of high heat-flux thermal management schemes, *IEEE Transactions on Components and Packaging Technologies*, 24, 122-141
8. S. Osher and R. Fedkiw (2003), *Level set methods and dynamic implicit surfaces*, Springer, New York. *Applies Mathematical Sciences*: Vol. 153.

9. M. Sarkar and R.P. Selvam (2005), Droplet Velocity and Liquid to Vapor Density Ratio Effect on Heat Transfer in Spray Cooling, Report, Computational Mechanics Laboratory, BELL 4190 University of Arkansas, Fayetteville, AR 72701.
10. R. P. Selvam (1997), Computation of pressures on Texas Tech building using large eddy simulation, *J. Wind Engineering and Industrial Aerodynamics*, 67 & 68, 647-657.
11. R.P. Selvam and R. Ponnappan (2004), Numerical modeling of nucleation boiling in thin film and effect of droplet impact, *Proceedings of the 15<sup>th</sup> Annual Thermal & Fluids Analysis Workshop (TFAWS 2004)*, Pasadena, CA, Aug. 30<sup>th</sup> to Sep. 3<sup>rd</sup>.
12. R.P. Selvam, L.Lin and R. Ponnappan (2005a), Computational modeling of spray cooling: Current status and future challenges, *Proceedings of the Space Technology and Applications International Forum (STAIF 2005)*, Conference on Thermophysics in Microgravity, Albuquerque, NM, Feb. 13-17
13. R.P. Selvam, L. Lin and R. Ponnappan, (2005b), Direct Simulation of Spray Cooling: Effect of Vapor Bubble Growth and Liquid Droplet Impact on Heat Transfer, paper to be submitted for publication in *Journal of Heat Transfer*.
14. R.P. Selvam, S. Baskara, J.C. Balda, F. Barlow and A. Elshabini (2005c), Computer Modeling of Liquid Droplet Impact on Heat Transfer During Spray Cooling, *Proceedings of the 2005 ASME Summer Heat Transfer Conference*, paper no: HT2005-72569, San Francisco, CA, July 17-22.
15. J.A. Sethian (1999), *Level set methods and fast marching methods: Evolving interfaces in computational geometry, fluid mechanics, computer vision and materials science*, Cambridge University Press, Cambridge, UK.
16. G. Son, and V.K. Dhir (1998), Numerical simulation of film boiling near critical pressures with a level set method, *Journal of Heat Transfer*, 120, 183-192.
17. G. Son, N. Ramanujapu and V.K. Dhir (2002), Numerical simulation of bubble merger process on a single nucleation site during pool nucleate boiling, *Journal of Heat Transfer*, 124, 51-62.
18. M. Sussman, P. Smereka and S. Osher, (1994), A level set approach for computing solutions to incompressible two-phase flow, *J. Comp. Physics*, 114,146-159.
19. J. Yang, L.C. Chow and M.R. Paris (1996), Nucleate boiling heat transfer in spray cooling, *Journal of Heat Transfer*, 118, 668-671

## Contact

Dr. R. Panneer Selvam is currently a Professor of Civil Engineering and Director of Computational Mechanics Laboratory at the University of Arkansas. He is currently involved in thermal management of high heat flux for power electronic devices and electronic packaging, multiphase flow modeling of spray cooling in micro and macro gravity environment, atomistic modeling of thermal effects on grain boundary of nanocrystals, multiscale (nano & continuum mechanics) modeling for material science and computer modeling tornado effects on buildings. He has been involved in computer modeling for engineering and science applications for the last 20 years. He has worked on computer modeling of bridge flutter, panel flutter, pollutant transport around buildings, thermal and creep stress analysis of electronic packaging and fracture mechanics. He has published more than 150 research papers. He is a member of ASCE, AAWE and IMAPS. Email: [rps@engr.uark.edu](mailto:rps@engr.uark.edu), tel: 479-575-5356, fax: 479-575-7168.

Ms. Mita Sarkar is Ph.D. student in the Computational Mechanics Laboratory, at the University of Arkansas. She is involved in the thermal management and modeling of spray cooling.

Dr. Ponnappan is currently a Senior Mechanical Engineer and in-house Program Manager at Air Force Research Laboratory. He is actively involved in thermal management research and manages several AF funded programs including spray cooling and has made significant contributions in heat pipes and electronics cooling area. He has pursued thermal technologies for over 25 years; organized and chaired sessions in many technical conferences. He co-invented the concept of double-wall artery wick and explored gas-loaded liquid metal heat pipe startup and developed facilities at WPAFB for testing heat pipes in high 'g' centrifuge. His 'rotor cooling structure' patent has been licensed by an aircraft alternator manufacturer for improving bearing life and reliability. Dr. Ponnappan pioneered research on the high speed rotating heat pipe. He is author of five US patents and over a hundred research publications. He is an Associate Fellow of AIAA and Fellow of ASME-International. Email: [rengasamy.ponnappan@wpafb.af.mil](mailto:rengasamy.ponnappan@wpafb.af.mil) Tel: 937-255-2922; DSN: 785-2922; Fax: 937-904-7114.

## Nomenclature

$c_p$	specific heat at constant pressure
$Fr$	Froude number
$g$	gravity vector
$H$	step function
$h$	grid spacing
$h_{fg}$	latent heat of evaporation
$Ja$	Jacob number = $c_{pl} \Delta T / h_{fg}$
$k$	thermal conductivity
$l_r$	characteristic length $\sqrt{\sigma / g(\rho_l - \rho_v)}$
$\mathbf{m}$	mass flux vector
$Nu$	Nusselt number $q l_r / (\Delta T k_l)$
$p$	pressure
$Pe$	Peclet number = $\rho_l u_r l_r c_{pl} / k_l$
$Pr$	Prandtl number = $c_{pl} \mu_l / k_l$
$q$	heat flux
$Re$	Reynolds number = $\rho_l u_r l_r / \mu_l$
$T$	temperature
$T^*$	dimensionless temperature $(T - T_{sat}) / (T_w - T_{sat})$
$\Delta T$	temperature difference $T_w - T_{sat}$
$t$	time
$t_r$	characteristic time $l_r / u_r$
$\mathbf{u}$	velocity vector (u,v)
$\mathbf{u}_{int}$	interface velocity vector
$u_r$	characteristic velocity $\sqrt{gl_r}$
$We$	Weber number = $\rho_l u_r^2 l_r / \sigma$
$\alpha$	thermal diffusivity
$\kappa$	interfacial curvature
$\mu$	dynamic viscosity

$\rho$  density  
 $\sigma$  surface tension  
 $\varphi$  level set function

*Subscripts*

int interface  
l, v liquid , vapor  
sat, w saturation, wall